# MSC Apex® | Structures

## Computational Parts Based Structural Analysis

Overview

MSC Apex Structures is an integrated module of Finite Element solver, which provides users the access to linear (and very soon, nonlinear) structural analysis. From now, Apex supports four types of linear analysis, linear statics, linear buckling, normal modes, and frequency response.

MSC Apex Structures packages an intuitive user interface for scenario definition, analysis readiness check, and integrated solver. This solution is unique in that it combines computational parts and assemblies based technology with a generative framework, which enables interactive and incremental analysis.

The integration of the user interface with solver gives users a unique ability to interactively and incrementally validate and solve FEM models. At the user's demand, a series of solver checks can be run against individual parts and assemblies, and the model diagnostics are reported in the Analysis Readiness panel. This Incremental Validation and Solution philosophy is a creative and intelligent revolution to the very time-consuming traditional workflow where pre/post processor and solver are separate.
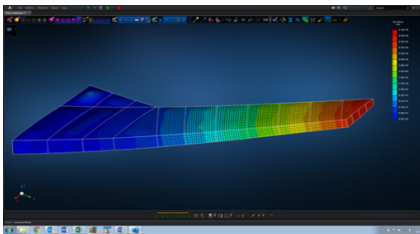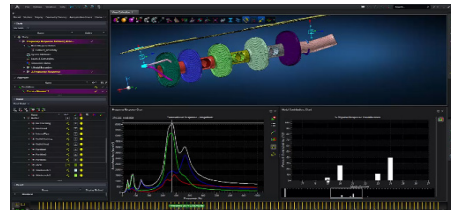

Linear Statics
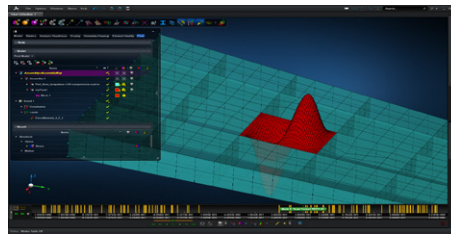
Linear Buckling
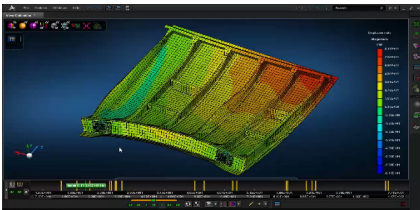
Normal Modes

Frequency Response

Apex supports "Apex-Nastran-Apex" workflow, which provides users multiple workflow scenarios and allows them to take advantage of one or both solvers:

- Scenario 1 – External MSC Nastran Solution - Many existing Nastran customers, will determine to use MSC Nastran as an external solver, due to in-house process and/or client requirements.

- Scenario 2 – Integrated Apex Structures Solution to Support External MSC Nastran Solution - The integrated solver can be used for incremental building and validation of FEA models as they are developed. Once verified as working representations, the run-ready FEA models can be solved externally with MSC Nastran, for subsequent result exploration through Apex.

- Scenario 3 – Embedded Apex Structures Solution – In the case where an MSC Nastran solution is not mandatory, the user can take advantage of the full capability of the embedded Apex solver.

Capabilities

- **Generative Framework**
  - Update Geometry, Mesh, Material, Property and Behaviors, Glue: mesh-dependent connection, Load, and Boundary Conditions, Scenarios and Results automatically with changes to the model

- **Incremental Validation**
  - Geometry specific (Part, Sub-assembly, Assembly)
  - Generative Analysis Readiness for mesh, materials, properties, LBCs, interactions, and simulation settings to guide the user on quickly preparing run-ready models

- **Incremental Solve**
  - Perform linear structural analysis with embedded Computational Parts and Assemblies based philosophy

- **Linear Structural Analysis**
  - Linear Statics
  - Normal Modes
  - Linear Buckling
  - Frequency Response

- **Results Exploration**
  - Use a powerful probe tool to identify critical displacements and stresses
  - Animate structural deformations and contour changes
  - View and switch interactively among multiple normal modes via modes navigator
  - Utilize a Results Manager to explore analysis
  - results by study, entities, and analysis types
  - Transform results to Cartesian, cylindrical, and spherical coordinate systems
  - View colorful fringe contours of physical quantities of interest, such as displacements, stresses, strains, etc.
  - Demonstrate vector plots of displacements, applied loads, constraint reactions, and more variables
  - Create sensors and monitor responses at specific locations, such as displacements and stresses
  - Display results in XY plots
  - Explore results under specific failure criteria

- **Study Manager**
  - Manage multiple scenarios (model representations, output requests, analysis type)
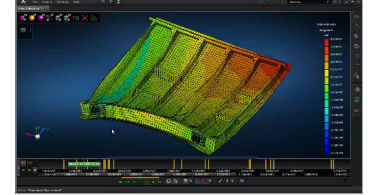
Structural Analysis Workflow

1   **Set Model and Analysis Context**

Define the analysis type
and a subset of parts
and assemblies to be the
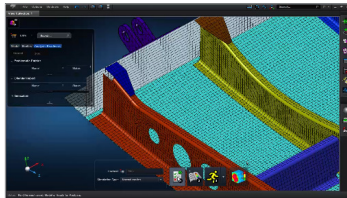context of evaluation.
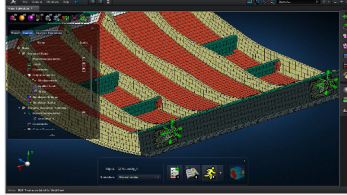
2   **Validate Models Before Analysis**

Use the integrated analysis
readiness tool to validate
the context has valid model
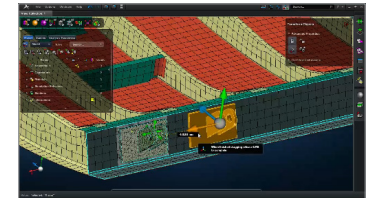representations for the
chosen analysis type.

3   **Join Dissimilar Meshes Rapidly**

Reduce the need to align
nodes across mesh parts
using mesh independent
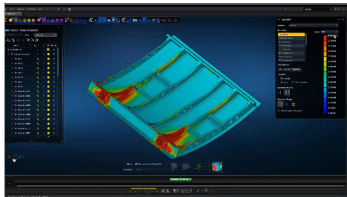glue technology.

4   **Make Generative Changes**

Track the status and manage
the update of downstream
updates whose parent
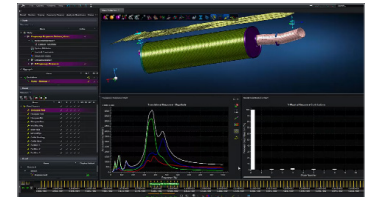has been modified.

5   **Generate and Visualize Results for
Linear Statics**

Define a linear static scenario
and execute the integrated
solver methods to generate
results interactively.

6   **Generate and Visualize Results for
Frequency Response**

Perform frequency response
analysis & use a results
exploration toolset to develop
solutions to mitigate and
control structural vibrations.

Productivity Gains

MSC Software®